AC-CBS-Based Partitioned Semi-Implicit Coupling Algorithm for Fluid-Structure Interaction Using Stabilized Second-Order Pressure Scheme

Tao He¹,², Kai Zhang³,⁴ and Tong Wang¹,*

¹ Department of Civil Engineering, Shanghai Normal University, Shanghai 201418, China.
² School of Engineering, University of Birmingham, Birmingham B15 2TT, UK.
³ School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China.
⁴ Department of Civil Engineering, Graduate School of Urban Innovation, Yokohama National University, Yokohama 2408501, Japan.

Received 14 July 2016; Accepted (in revised version) 18 October 2016

Abstract. We analyze in this paper the pressure splitting scheme of a partitioned semi-implicit coupling algorithm for fluid-structure interaction (FSI) simulation. The semi-implicit coupling algorithm is developed on the ground of the artificial compressibility characteristic-based split (AC-CBS) scheme that serves not only for the fluid subsystem but also for the global FSI system. As the dual-time stepping procedure recommended for quasi-incompressible flows is incorporated into the implicit coupling stage, the fluctuating pressure may be unusually susceptible to the AC coefficient. Moreover, it is not trivial to devise an optimal AC formulation for pressure estimation. Instead, we consider a stabilized second-order pressure splitting scheme in the AC-CBS-based partitioned semi-implicit coupling algorithm. Computer simulation of a benchmark FSI experiment demonstrates that good agreement is exposed between the available and present data.

AMS subject classifications: 35Q30, 65M22, 65M60, 74F10, 76M10
Key words: Fluid-structure interaction, semi-implicit coupling, CBS, artificial compressibility, second order.

1 Introduction

The numerical simulation of fluid-structure interaction (FSI) has long attracted vast interests from research community because of its scientific and practical importance. In

*Corresponding author. Email addresses: txh317@bham.ac.uk (T. He), kai-zhang-kf@ynu.jp (K. Zhang), tongwang@shnu.edu.cn (T. Wang)
consideration of FSI, people can moderately handle challenging problems, such as wind-
induced vibrations of a skyscraper or super suspension bridge in civil engineering. Al-
though an FSI problem can be solved in a monolithic manner [1, 2], partitioned solution
technique under the arbitrary Lagrangian-Eulerian (ALE) description [3, 4] seems a pre-
vailing strategy. Interested readers can refer to recent review articles [5, 6] for the repre-
sentative numerical methods of FSI.

Apart from traditional explicit and implicit approaches, partitioned semi-implicit cou-
pling algorithm has been proposed over the last decade. Fernández et al. [7] first pro-
posed a projection semi-implicit coupling scheme for simulating FSI problems with strong
added-mass effect [8–10]. The basis of this semi-implicit concept depends on the classical
Chorin-Témam splitting [11, 12] which naturally offers an explicit-implicit treatment for
the FSI resolution. In particular, the ALE-advection-diffusion step is explicitly treated
with the predicted fluid mesh while the projection step is implicitly coupled with the
structural motion on the previously frozen fluid mesh. Theoretical analysis indicated
that the semi-implicit coupling algorithm exhibits the enhanced computational efficiency
without affecting the stability condition significantly when compared to the implicit cou-
pling algorithm [7]. Following this idea, a number of semi-implicit coupling methods
have been presented over the past years. Quaini and Quarteroni [13] invented a semi-
implicit coupling scheme on the ground of algebraic fractional step method. This is the
first time that the algebraic fractional step method is applied to FSI. Unlike differential
splitting, the algebraic fractional step method requires no auxiliary boundary conditions
for differential subsystems divided by the original problem. Badia et al. [14] introduced
the inexact block-LU factorization into several semi-implicit coupling schemes and dis-
cussed their own performances. Nitsche-based and Robin-based semi-implicit coupling
schemes, as well as a couple of variants, were proposed for an incompressible viscous
fluid interplaying with a thin-walled solid [15]. Better stability properties were achieved
through these hybrid interface conditions. Astorino et al. [16] performed a convergence
analysis for the projection-based semi-implicit coupling scheme [7] in a simplified FSI
system. The error of time discretization was proved to be at least $\sqrt{\Delta t}$ in this scheme.
Fernández [17] presented a comprehensive review for numerical simulation of blood
flows in large arteries involving explicit, semi-implicit and implicit coupling schemes.
The projection-based partitioned semi-implicit coupling strategy was successfully ap-
plied to a three-dimensional idealized abdominal aortic aneurysm for sequential param-
eter estimation by using reduced-order unscented Kalman filter [18].

A new partitioned semi-implicit coupling method was developed by the authors on
the basis of the characteristic-based split (CBS) scheme, a general algorithm for fluid dy-
namics [19–21]. The resulting algorithm was termed the CBS-based partitioned semi-implicit
coupling method. It is seen from [19] that a set of additional ordinary differential equations
need to be solved on the constructed fluid-structure interface. Unlike [19], the traditional
interface conditions were enforced in [20, 21]. The first author’s research work was briefly
summarized in [20] by comparing different partitioned coupling schemes for two large-
displacement FSI problems. Ref. [21] provided the details on carrying out the smoothed
finite element technique for the structural part of the partitioned solution strategy. Most recently, He and Zhang [22, 23] developed the improved interface coupling conditions in association with the CBS-based partitioned semi-implicit coupling method.

Instead of adopting the Chorin-Témam splitting, a handful of partitioned semi-implicit coupling schemes solve the fluid momentum equation monolithically. Sy and Murea [24, 25] solved a least squares problem on the interface so as to ensure the continuity of the velocity and stress in a semi-implicit way. The explicit coupling phase only predicts the fluid mesh while the fluid and solid resolutions are deposited within the implicit iterations. Breuer et al. [26] independently designed a partitioned semi-implicit predictor-corrector coupling scheme to study an FSI benchmark problem in turbulent flows. The mesh adaptation is repeated at the corrector step (i.e. the implicit coupling step), differing from [24, 25].

Despite that substantial accomplishments have been reported in the cited literature, the projection-based semi-implicit coupling algorithm remains far away from being perfect. A monolithic formulation of the fluid projection step and the structural motion is cast at the expense of iteratively solving a set of linearized algebraic equations in [7]. The formulation is very similar to that applied to the incompressible Navier-Stokes (NS) equations as usual. Subsequently, a coupled pressure-interface system is established after discretizing and linearizing the fluid-structure system, invoking the pressure-interface correction method and fluid-structure Yoshida method [14]. As a matter of fact, the semi-implicit formulation is processed into the compact form at partly monolithic level. To construct the fraction-step-type partitioned semi-implicit coupling scheme, the first author embedded a mass source term (MST) [27] in the pressure Poisson equation (PPE) within the CBS scheme for the elements adhering to the fluid-structure interface [20–23]. The MST is rigorously derived in the context of three-node triangular (T3) finite element. Based on these observations, the following deficiencies of the projection-based partitioned semi-implicit coupling method are encountered

- Loss of the fractional-step modularity;
- Complex mathematical management and more numerical expenditure resulting from an algebraic system;
- Dependence on finite element while maintaining the fractional-step calculation.

As a consequence, the goal of this study consists in proposing an alternative approach using the artificial compressibility (AC) method [28]. To actualize the AC-type partitioned semi-implicit coupling algorithm, we must accommodate the PPE at the implicit coupling stage and minimize numerical efforts on a general finite element mesh. We consider here implicit subiterations between the fluid projection step and the structural motion at each time step as the pseudo-time iterations. A suitable AC coefficient is determined to refrain from the inferior results. The matrix-free computation of the decoupled FSI system is also pursued.

The remainder of this paper is organized as follows. In Section 2 the governing equations are depicted for all components. The details of the proposed semi-implicit coupling
scheme are presented in Section 3. Numerical example is investigated in Section 4. Concluding remarks are drawn in the final section.

2 Mathematical models

2.1 Fluid flows with moving boundaries

Let $\Omega^F_t \subset \mathbb{R}^2$ and $(0, T)$ be the fluid and temporal domains, respectively. $\Omega^F_t$ is bounded by $\Gamma^F_t$ which is decomposed into three complementary subsets, i.e., the Dirichlet-type boundary $\Gamma^D_F$, the Neumann-type boundary $\Gamma^N_F$ and the fluid-structure interface $\Sigma$. The spatial and temporal coordinates are denoted by $x$ and $t$. The unsteady NS equations in the arbitrary Lagrangian-Eulerian (ALE) description governing the fluid flows on a moving domain read as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad \text{on} \quad \Omega^F_t \times (0, T),$$ (2.1)

$$\rho \left( \frac{\partial u}{\partial t} + c \cdot \nabla u - f \right) - \nabla \cdot \sigma = 0 \quad \text{on} \quad \Omega^F_t \times (0, T),$$ (2.2)

where the primitive variables are the fluid velocity $u$ and the pressure $p$, $\rho$ denotes the fluid density, $c = u - w$ is the convective velocity, $w$ is the mesh velocity, $f$ represents the body force, $\sigma$ is the fluid stress tensor and $\nabla$ means the gradient operator.

Eq. (2.1) states the general form of the continuity equation or mass conservation, which for the incompressible fluid simplifies to

$$\nabla \cdot u = 0.$$ (2.3)

For the slightly compressible or quasi-incompressible fluid flows, the AC is employed to improve the conditioning of the resulting hyperbolic set of the NS equations. The AC approach was first proposed by Chorin [28] for the steady state flows and later extended by Turkel [29] in the unsteady case. As the fluid density $\rho$ is independent of spatial coordinates $x$ in usual, the continuity equation (2.1) is altered by inserting a pressure time derivative such that

$$\frac{1}{a^2} \frac{\partial p}{\partial t} + \rho \nabla \cdot u = 0,$$ (2.4)

where $a$ designates the AC coefficient or speed of sound ($a \rightarrow \infty$ for the incompressible fluid flows).

The constitutive equation for a Newtonian fluid is written as

$$\sigma = -pI + 2\mu \epsilon \quad \text{and} \quad \epsilon = \frac{1}{2} \left( \nabla u + (\nabla u)^T \right),$$ (2.5)

where $I$ indicates the identity matrix, $\mu$ is the fluid viscosity, $\epsilon$ is the rate-of-strain tensor and superscript $T$ indicates transpose.
The fluid problem is completed by prescribing boundary and initial conditions below
\[ u = g \text{ on } \Gamma_D^F, \quad \sigma \cdot n = h \text{ on } \Gamma_N^F, \]
\[ u(x, 0) = u^0, \quad p(x, 0) = p^0 \text{ on } \Omega_0^F, \]
where \( n \) is the unit outward normal of \( \Gamma_N^F \).

In order to facilitate the fluid simulation, the following dimensionless scales are defined
\[ x^* = \frac{x}{D}, \quad t^* = \frac{tU}{D}, \quad u^* = \frac{u}{U}, \quad a^* = \frac{a}{U}, \quad c^* = \frac{c}{U}, \quad p^* = \frac{p}{\rho U^2}, \quad f^* = \frac{fD}{U^2} \]
based on the free stream velocity \( U \) and the characteristic length \( D \). By employing these scales and dropping all asterisks, the dimensionless version of the quasi-incompressible Navier-Stokes equations is obtained as follows
\[ \frac{1}{a^2} \frac{\partial p}{\partial t} + \nabla \cdot u = 0 \text{ on } \Omega_t^F \times (0, T), \tag{2.6} \]
\[ \frac{\partial u}{\partial t} + c \cdot \nabla u - \nabla \cdot \sigma - f = 0 \text{ on } \Omega_t^F \times (0, T). \tag{2.7} \]

In the above dimensionless equations
\[ \sigma = -pI + \frac{1}{Re} \left( \nabla u + (\nabla u)^T \right), \tag{2.8} \]
where \( Re = \rho UD / \mu \) is the Reynolds number.

A well-established NS solver is the so-called artificial compressibility characteristic-based split (AC-CBS) scheme [30, 31] that combines the characteristic Galerkin method [32, 33] with the fractional-step method [11, 12] under the umbrella of AC. The characteristic-based process reduces oscillations via higher-order time stepping in the convection-dominated flows, while the fractional-step solution procedure stabilizes the pressure. As we know, the original fractional-step procedure introduces the first-order pressure splitting error into the momentum equation if the pressure gradient term is completely removed from the equation. The inclusion of this term guarantees the second-order pressure splitting error. At this point the increased accuracy may engender the reduced pressure stability, especially for steady-state problems [34, 35]. More acute sensitivity can be found on pressure boundaries. In what follows, a second-order AC-CBS scheme with the stabilized pressure gradient projection (SPGP) technique [34, 36, 37] will be elaborated for the fluid problem.

We define an auxiliary equation
\[ q - \nabla p = 0, \tag{2.9} \]
where \( q \) is the auxiliary variable, and then modify the continuity equation (2.6) as
\[ \frac{1}{a^2} \frac{\partial p}{\partial t} + \nabla \cdot u + \chi \nabla \cdot q - \chi \nabla^2 p = 0, \tag{2.10} \]
with $\chi$ denoting a stabilization parameter \cite{37, 38}. The parameter $\chi = \Delta t / 4$ is adopted here according to \cite{35, 37}.

The temporal discrete versions of Eqs. (2.10), (2.7) and (2.9) may be written as

\[
\left( \frac{1}{\Delta t} \right) \frac{p^{n+1} - p^n}{\Delta t} + \nabla \cdot u^{n+1} + \chi \nabla \cdot q^n - \chi \nabla^2 p^{n+1} = 0, \tag{2.11}
\]

\[
\frac{u^{n+1} - u^n}{\Delta t} = -c^n \cdot \nabla u^n - \nabla p^n + \nabla p^n + \frac{1}{\Re} \nabla^2 u^n + f^n, \tag{2.12}
\]

\[
q^{n+1} - \nabla p^{n+1} = 0 \tag{2.13}
\]

where superscripts $n$ and $n+1$ denote the $n$th and $(n+1)$th time slices, respectively, and $\Delta t$ is the time step. Eq. (2.11) implies that the pressure gradient term is explicitly treated.

Following the CBS procedure, an intermediate velocity $\tilde{u}$ is introduced to split Eq. (2.12) into

\[
\frac{\tilde{u} - u^n}{\Delta t} = -c^n \cdot \nabla u^n - \nabla p^n + \frac{1}{\Re} \nabla^2 u^n + \frac{\Delta t}{2} c^n \cdot \nabla (c^n \cdot \nabla u^n + \nabla p^n), \tag{2.14}
\]

\[
\frac{u^{n+1} - \tilde{u}}{\Delta t} = -\nabla (p^{n+1} - p^n) + \frac{\Delta t}{2} c^n \cdot \nabla^2 (p^{n+1} - p^n), \tag{2.15}
\]

where the body force vector and the third-order terms are neglected. Taking the divergence of Eq. (2.15) and expanding the semi-discrete form of Eq. (2.11) at the next time level yields

\[
\left( \frac{1}{\Delta t} \right) \frac{p^{n+1} - p^n}{\Delta t} = -\nabla \cdot \tilde{u} + \Delta t \nabla^2 (p^{n+1} - p^n) - \chi \nabla \cdot q^n + \chi \nabla^2 p^n, \tag{2.16}
\]

where the third-order terms are discarded again. As is discriminated by superscript $m$ in this equation, the implicit dependence on $p^{n+1}$ on the right-hand side is explicitly altered so as to ease the anticipative matrix-free calculation.

After the standard Galerkin finite element discretization, the main steps of the stabilized second-order AC-CBS scheme within each time interval are summarized as follows

- Calculate the intermediate velocity $\tilde{u}$ using Eq. (2.14);
- Update the pressure $p^{n+1}$ based on Eq. (2.16);
- Establish the corrected velocity $u^{n+1}$ via Eq. (2.15);
- Assess the auxiliary variable $q^{n+1}$ through Eq. (2.13).

It is noticed that no boundary condition is enforced for the intermediate velocity field, as treated in compliance with \cite{39, 40}.

The AC-CBS scheme is technically a non-projection variant as the incompressibility constraint is not fulfilled. Nevertheless, it is rather similar to the classical Chorin-Témam projection method and offers the footstone of our semi-implicit coupling method. As a result, the terminology projection or fluid projection step is still accepted in this study.
Furthermore, slight adjustments are bound to be made in order to accommodate the proposed partitioned semi-implicit coupling algorithm. It is emphasized that in Eq. (2.6) we make the pseudo-time step $\Delta \tau$ equal to the physical time step $\Delta t$. The traditional dual-time stepping procedure, which possibly brings about triple loops in partitioned subiterative coupling scheme, is thus avoided for the enhanced efficiency.

Further to the AC coefficient $a$, it is supposed to include the pressure solution within the implicit coupling phase. For this reason, this parameter must be defined on the foundation of the fluid velocity. It is natural to adopt the criterion proposed by Nithiarasu [41, 42]. Nonetheless, our computer experiments indicate that Nithiarasu’s AC formulation underestimates numerical results in the current scenario, and suggest the locally varying value [43]

$$a^2 = \max(\varepsilon^2, 2.5|u|^2),$$

for better performance, where $\varepsilon=1$. In addition, the following inequality may be satisfied

$$a^2 \gg \left(1 + \frac{4D}{Re}\right)^2 - 1,$$

for low-speed incompressible flows [44].

### 2.2 Structural motion

We consider $\Omega^S \subset \mathbb{R}^2$ a structural domain with the boundary $\Omega^S_t$. A rigid structure immersed in a fluid is modeled as a spring-damper-mass system under the isotropic assumption. $d = [d_1, d_2, \theta]^T$ signifies the structural displacement where all components are defined at the center of gravity $G$, and subscripts 1, 2 and $\theta$ designates the horizontal, vertical and rotational directions (Fig. 1). Along with proper boundary and initial conditions, the equation of structural motion is formulated in the Lagrangian kinematics below

$$\begin{bmatrix} m_1 & m_2 \\ m_2 & m_0 \end{bmatrix} \ddot{d} + \begin{bmatrix} c_1 \\ c_2 \end{bmatrix} \dot{d} + \begin{bmatrix} k_1 \\ k_2 \end{bmatrix} d = P,$$

where the dot illuminates the time derivative, $m_i, c_i$ and $k_i$ stand for the mass, damping and stiffness of the structure, $P = \{F_D, F_L, F_M\}^T$ is the fluctuating fluid force, $F_D, F_L$ and $F_M$ mean the drag, lift and pitching moment, respectively.

The compatibility condition must be satisfied between the center of gravity $G$ and the surface point $P$, see [45] for details. As pictured in Fig. 1, the geometric relation between $d$ and $d^P$ is written in the component form as follow

$$\begin{bmatrix} d_1^P \\ d_2^P \end{bmatrix} = \begin{bmatrix} d_1 \\ d_2 \end{bmatrix} + \begin{bmatrix} \cos \theta - 1 \\ \sin \theta \end{bmatrix} \begin{bmatrix} x_1^P \\ x_2^P \end{bmatrix},$$

where $d^P$ is the displacement of $P$ and $\{x_1^P, x_2^P\}^T$ is the coordinates of $P$. 
By differentiating Eq. (2.20) with respect to $t$, the velocity relation is expressed as
\[
\begin{bmatrix}
\dot{d}_1^P \\ \dot{d}_2^P 
\end{bmatrix} = \begin{bmatrix}
\dot{d}_1 \\ \dot{d}_2 
\end{bmatrix} + \theta \begin{bmatrix}
-\sin \theta & -\cos \theta \\ \cos \theta & -\sin \theta 
\end{bmatrix} \begin{bmatrix}
x_1^P \\ x_2^P 
\end{bmatrix} = \begin{bmatrix}
1 & 0 \\ 0 & 1 
\end{bmatrix} \begin{bmatrix}
-\dot{L}_1^P \\ \dot{L}_1^P 
\end{bmatrix} \begin{bmatrix}
\dot{d}_1 \\ \dot{d}_2 
\end{bmatrix},
\]
(2.21)
where $\dot{L}_1^P = x_1^P \cos \theta - x_2^P \sin \theta$ and $\dot{L}_2^P = x_2^P \sin \theta + x_1^P \cos \theta$ are the angle-dependent coefficients.

Similarly, the following acceleration relation is obtained by differentiating Eq. (2.21) in terms of time
\[
\begin{bmatrix}
\ddot{d}_1^P \\ \ddot{d}_2^P 
\end{bmatrix} = \begin{bmatrix}
\dot{d}_1 \\ \dot{d}_2 
\end{bmatrix} + \theta \begin{bmatrix}
-\sin \theta & -\cos \theta \\ \cos \theta & -\sin \theta 
\end{bmatrix} \begin{bmatrix}
x_1^P \\ x_2^P 
\end{bmatrix} + \theta^2 \begin{bmatrix}
-\cos \theta & \sin \theta \\ -\sin \theta & -\cos \theta 
\end{bmatrix} \begin{bmatrix}
x_1^P \\ x_2^P 
\end{bmatrix} + \dot{\theta} \begin{bmatrix}
0 & 0 \\ -L_2^P & L_1^P 
\end{bmatrix} \dot{\theta}.
\]
(2.22)

The dimensionless scales
\[
\begin{align*}
x^* &= \frac{x}{D}, & t^* &= \frac{t}{U}, & d_1^* &= \frac{d_1}{D}, & d_2^* &= \frac{d_2}{D}, \\
C_D &= \frac{2F_D}{\rho U^2 D}, & C_L &= \frac{2F_L}{\rho U^2 D}, & C_M &= \frac{2F_M}{\rho U^2 D^2}, \\
m_1^* &= \frac{m_1}{\rho D^2}, & m_2^* &= \frac{m_2}{\rho D^2}, & m_\theta^* &= \frac{m_\theta}{\rho D^4}
\end{align*}
\]
and the reduced parameters
\[
\begin{align*}
\xi_1 &= \frac{c_1}{2\sqrt{m_1 k_1}}, & \xi_2 &= \frac{c_2}{2\sqrt{m_2 k_2}}, & \xi_\theta &= \frac{c_\theta}{2\sqrt{m_\theta k_\theta}}.
\end{align*}
\]
are computed to nondimensionalize Eq. (2.19), where the drag coefficient \( C_D \), the life coefficient \( C_L \) and the moment coefficient \( C_M \) are the dimensionless applied forces, the mass ratio \( m_i^* \) is the dimensionless mass, \( \zeta_i \) is the damping ratio, \( f_{Ri} \) is the reduced natural frequency, and \( f_{Ni} \) is the natural frequency. By considering the above variables without superscript asterisks, the dimensionless equation of structural motion is written as

\[
\ddot{d} + 4\pi \left[ \begin{array}{c} f_{R1} \xi_1 \\ f_{R2} \xi_2 \\ f_{R\theta} \xi_\theta \end{array} \right] d + 4\pi^2 \left[ \begin{array}{c} (f_{R1})^2 \\ (f_{R2})^2 \\ (f_{R\theta})^2 \end{array} \right] \dot{d} = \begin{cases} C_D \\ \frac{2m_1^*}{C_L} \\ \frac{2m_2^*}{C_M} \end{cases} \right].
\tag{2.23}
\]

The widespread availability of step-by-step time integration algorithms is seen in the computational analyses of structural dynamics. Apart from the famous Newmark-\( \beta \) method [46], other time marching methods have been proposed, to name but a few, the Hibert-Hughes-Taylor-\( \alpha \) method [47], the Generalized-\( \alpha \) method [48], the composite implicit time integration method [49] and the time discontinuous Galerkin method [50–52]. These time integration algorithms have been proven to surpass the traditional Newmark method. In this paper, the structural equation is integrated in time with the Generalized-\( \alpha \) method [48]. To this end, the semi-discrete Eq. (2.23) is applied to a general mid-point within the time interval, implying that the following modified equation holds

\[
M \ddot{d}^{n+1} - \alpha_m \ddot{d}^{n} + C \ddot{d}^{n+1} - \alpha_f \ddot{d}^{n} + K \ddot{d}^{n+1} - \alpha_f \ddot{d}^{n} = F^{n+1} - \alpha_f F^{n},
\tag{2.24}
\]

where

\[
\begin{align*}
\ddot{d}^{n+1} - \alpha_m \ddot{d}^{n} & = (1 - \alpha_m) \ddot{d}^{n+1} + \alpha_m \ddot{d}^{n}, \\
\ddot{d}^{n+1} - \alpha_f \ddot{d}^{n} & = (1 - \alpha_f) \ddot{d}^{n+1} + \alpha_f \ddot{d}^{n}, \\
\ddot{d}^{n+1} - \alpha_f \ddot{d}^{n} & = (1 - \alpha_f) \ddot{d}^{n+1} + \alpha_f \ddot{d}^{n}, \\
F^{n+1} - \alpha_f F^{n} & = (1 - \alpha_f) F^{n+1} + \alpha_f F^{n}.
\end{align*}
\tag{2.25}
\]

To set \( d^{n+1} \) as the single unknowns in Eq. (2.24), the Newmark approximations [46] to the acceleration and velocity at new time step are stated as

\[
\begin{align*}
\ddot{d}^{n+1} = \frac{1}{\beta \Delta t^2} (d^{n+1} - d^n) - \frac{1}{\beta \Delta t} \ddot{d}^n - \frac{1 - 2\beta}{2\beta} \ddot{d}^n, \\
\dot{d}^{n+1} = \frac{\gamma}{\beta \Delta t} (d^{n+1} - d^n) - \frac{\gamma - \beta}{\beta} \dot{d}^n - \frac{\gamma - 2\beta}{2\beta} \Delta t \dot{d}^n.
\end{align*}
\tag{2.26}
\]

\[
\begin{align*}
\ddot{d}^{n+1} = \frac{1}{\beta \Delta t^2} (d^{n+1} - d^n) - \frac{1}{\beta \Delta t} \ddot{d}^n - \frac{1 - 2\beta}{2\beta} \ddot{d}^n, \\
\dot{d}^{n+1} = \frac{\gamma}{\beta \Delta t} (d^{n+1} - d^n) - \frac{\gamma - \beta}{\beta} \dot{d}^n - \frac{\gamma - 2\beta}{2\beta} \Delta t \dot{d}^n.
\end{align*}
\tag{2.27}
\]
Accordingly, the generalized mid-point acceleration and velocity are given by
\[
\ddot{d}_{n+1} = \frac{1 - \alpha_m}{\beta \Delta t^2} (d_{n+1} - d^n) - \frac{1 - \alpha_m}{\beta \Delta t} d^n - \frac{1 - \alpha_m - 2\beta}{2\beta} d^n, \tag{2.28}
\]
\[
\dot{d}_{n+1} = \frac{(1 - \alpha_f)\gamma}{\beta \Delta t} (d_{n+1} - d^n) - \frac{(1 - \alpha_f)\gamma - \beta}{\beta} d^n - \frac{(\gamma - 2\beta)(1 - \alpha_f)}{2\beta} \Delta t \ddot{d}^n. \tag{2.29}
\]

According to Chung and Hulbert [48], the time integration parameters \( \beta, \gamma, \alpha_m \) and \( \alpha_f \) are defined as functions of the spectral radius \( \rho_\infty \) and the optimal expressions are specified by
\[
\beta = \frac{1}{4} (1 - \alpha_m + \alpha_f)^2, \quad \gamma = \frac{1}{2} - \alpha_m + \alpha_f, \quad \alpha_m = \frac{2\rho_\infty - 1}{\rho_\infty + 1}, \quad \alpha_f = \frac{\rho_\infty}{\rho_\infty + 1}, \tag{2.30}
\]
where \( 0 \leq \rho_\infty \leq 1 \) has to be chosen for the desired level of numerical dissipation. \( \rho_\infty = 0.1 \) is specified in accordance with Dettmer and Perić [53] throughout this paper.

### 2.3 Mesh deformation method

An essential aspect of FSI computation is concerned with the ALE mesh deformation caused by the structural movement. Our mesh deformation method adopts a blend of moving submesh approach (MSA) [54] and the ortho-semi-torsional spring analogy method (OST-SAM) [55] so as to significantly depress the time consumption.

The basic idea behind MSA lies in putting a layer of sparse submesh (zones) over the fluid dynamic mesh (elements) which is then dynamically rearranged through specific interpolation formulae. A node on the fluid mesh is distinctively referred to as a point, whereas that on the MSA submesh is still called a node. A simply graphic representation is shown in Fig. 2. The principle of this technique is outlined below:

**Step 1:** Extract information of the mesh and submesh
**Step 2:** Collect all fluid points falling into each MSA zone
**Step 3:** Calculate interpolation formulae for each point in the zone
**Step 4:** Begin time loop
   1. Gain wall nodes’ displacement from the structural motion
   2. Invoke OST-SAM to assess interior nodes’ motion (skip this substep if no interior nodes appear);
   3. Update the submesh
   4. Interpolate the ALE mesh based on the new submesh
   5. Check the quality of zones and dynamic elements

**Step 5:** End time loop

According to Lefrançois [54], several keypoints should be noticed. Firstly, only the triangle is available for the zone and hence the resulting interpolation or mapping function is actually the shape function of T3 element. This seems to be the major limitation of
MSA. Despite that, the fluid element can adopt either a triangle or quadrangle. Secondly, the first three steps need to be implemented just once at initial state. Thirdly, either the absolute or relative displacement can be used in the approach. Finally, a capsule is adopted to encapsulate those complex structures whose geometries are composed of segments or curves. We can surely make use of more than one capsules if necessary.

A submesh without any interior nodes permits the immediate application of MSA (see Step 4.2). When interior nodes arise, the pseudo-structural equation of elastodynamics has to be dealt with by appropriate measures. MSA works in conjunction with OST-SAM in this case. The resulting quasi-static equilibrium equations are settled by the simple successive over-relaxation method [56] instead of the complex preconditioned solver. It is emphasized that MSA is far more economical than SAM since (1) MSA possesses the very simple interpolation functions; (2) MSA utilizes a fast interpolation process whereas SAM requires the massive iterations; (3) MSA demands the remarkably fewer iterations only if it has to.

Since MSA preserves the quality of ALE mesh topology quite well, there is no need to smooth nodes’ coordinates [27, 57]. In fact, MSA is a variant of the approach proposed by Liu et al. [58] who employed the Delaunay triangulation for structures of arbitrary profile. This technique is much simpler for those users who are not familiar with Delaunay graph tools.

2.4 Interface coupling conditions

In partitioned FSI calculation, the interplay between the fluid and structure is accomplished via separately enforcing the velocity continuity and traction equilibrium on the
interface $\Sigma$ as follows

$$u = \dot{d} \quad \text{and} \quad t^F = t^S,$$

where $t^F = \sigma^F \cdot n^S$ and $t^S = \sigma^S \cdot n^S$ are the fluid and structural tractions respectively, $n^S$ represents the unit outward normal of $\Sigma$ pointing from the structure to the fluid and $n^F = -n^S$.

The external force acting on a rigid body immersed in a fluid is a concentrated load vector. The stress equilibrium on $\Sigma$ thus becomes

$$\int_{\Sigma} t^F d\Gamma = \int_{\Sigma} t^S d\Gamma, \quad (2.32a)$$

$$\int_{\Sigma} \Delta x \times t^F d\Gamma = \int_{\Sigma} \Delta x \times t^S d\Gamma, \quad (2.32b)$$

where $\Delta x$ is the distance between the surface point and the center of gravity, see Fig. 1 for reference.

Also, the geometric continuity should be supplemented thanks to the dynamic mesh motion

$$x = d \quad \text{and} \quad w = \dot{d}. \quad (2.33)$$

3 Implementations of semi-implicit fluid-structure coupling

3.1 The proposed algorithm

The present solution strategy, called AC-CBS-based partitioned semi-implicit coupling algorithm, is suggested in the similar fashion of [7, 22, 23]. To set up a matrix-free solution process, we slightly regulate the time discretization of the second-order CBS scheme and lump mass matrices for all steps. A second-order structural predictor [59] is utilized to extrapolate the interface. The fixed-point iteration with Aitken’s $\Delta^2$ method [60, 61] is carried out for the implicit coupling part owing to its attractive simplicity and efficiency. In light of Badia and Codina [62], this technique has a good convergence for aeroelasticity. The procedure of the proposed algorithm is particularized below.

Step 1: Initialize all variables and set $\text{iter} = 0$

Step 2: Perform the explicit coupling phase

2.1: Extrapolate the position of the interface

$$\left(\bar{x}_\Sigma\right)^{n+1}_{\text{iter}} = x^n_{\Sigma} + \Delta t \left(\frac{3}{2}x^n_{\Sigma} - \frac{1}{2}x^{n-1}_{\Sigma}\right)$$

2.2: Rearrange the fluid mesh by MSA

2.3: Calculate the mesh velocity and other geometric quantities

$$w^{n+1}_{\text{iter}} = \frac{x^{n+1}_{\text{iter}} - x^n}{\Delta t}$$
2.4: Compute the intermediate velocity

\[ u_{iter}^{n+1} - u^n = \Delta t \left( -c^n \cdot \nabla u^n - \nabla p^n + \frac{1}{Re} \nabla^2 u^n + \frac{\Delta t}{2} c^n \cdot \nabla (c^n \cdot \nabla u^n + \nabla p^n) \right) \]

**Step 3:** Perform the implicit coupling phase

3.1: Set \( \text{iter} \leftarrow \text{iter} + 1 \)

3.2: Assess the AC coefficient \( a_{iter-1}^{n+1} \)

3.3: Update the fluid pressure

\[ \frac{1}{\Delta t} \left( p_{iter}^{n+1} - p^n \right) = -\Delta t \left( \nabla \cdot u_0^{n+1} - \Delta t \nabla^2 (p_{iter-1}^{n+1} - p^n) + \chi \nabla \cdot q_{iter-1}^{n+1} - \chi \nabla^2 p_{iter-1}^{n+1} \right) \]

3.4: Correct the fluid velocity

\[ u_{iter}^{n+1} - u_0^{n+1} = -\Delta t \left( \nabla (p_{iter}^{n+1} - p^n) - \frac{\Delta t}{2} c^n \cdot \nabla (p_{iter}^{n+1} - p^n) \right) \]

3.5: Evaluate the auxiliary variable

\[ q_{iter}^{n+1} - \nabla p_{iter}^{n+1} = 0 \]

3.6: Deduce the fluid load and pass it to the structure

3.7: Solve the structural equation

\[ \left( \frac{1 - \alpha_m}{\beta \Delta t^2} M + \frac{(1 - \alpha_f)\gamma}{\beta \Delta t} C + (1 - \alpha_f) K \right) d_{iter}^{n+1} = (1 - \alpha_f) F_{iter}^{n+1} + \alpha_f F^n + M \left( \frac{1 - \alpha_m}{\beta \Delta t^2} d^n + \frac{1 - \alpha_m}{\beta \Delta t} d^n + \frac{1 - \alpha_m - 2\beta}{2\beta} d^n \right) + C \left( \frac{(1 - \alpha_f)\gamma}{\beta \Delta t} d^n + \frac{(1 - \alpha_f)(\gamma - \beta)}{\beta} d^n + \frac{(1 - \alpha_f)(\gamma - 2\beta)}{2\beta} d^n \right) - K\alpha_f d^n \]

3.8: Estimate the interfacial residuals

\[ g_{iter} = \left| (x_{\Sigma})_{iter}^{n+1} - (\tilde{x}_{\Sigma})_{iter-1}^{n+1} \right| \]

3.9: Check the convergence and the maximum number of subiterations:
- if not convergent, then go ahead;
- otherwise, proceed to the next time step

3.10: Determine Aitken factor \( \lambda_{iter} \)
3.11: Relax the interface’s position

\[ (\tilde{x}_{\Sigma})_{n+1}^{iter} = \lambda_{iter} (x_{\Sigma})_{n+1}^{iter} + (1 - \lambda_{iter}) (\tilde{x}_{\Sigma})_{n+1}^{iter-1} \]

3.12: Calculate the mesh velocity for the velocity boundary condition

\[ (w_{\Sigma})_{n+1}^{iter} = \frac{(\tilde{x}_{\Sigma})_{n+1}^{iter} - x_{\Sigma}^{n}}{\Delta t} \]

3.13: Return

The flowchart of our algorithm is also displayed in Fig. 3. Seen from the above algorithm, the AC coefficient \( a \) formulated in terms of velocity serves a vehicle to exchange...
information between explicit and implicit stages. This is quite different from our earlier PPE approach [22, 23]. In other algorithms [62, 63], multiple criteria may be adopted to judge the convergence locally and globally. Since the dual-time stepping is merged with the implicit subiterations, the unique convergence criterion is specified here. Compared to the previous semi-implicit approaches, such as [7, 22, 23], the following merits of the present stabilize second-order AC-CBS-based partitioned semi-implicit coupling scheme are observed:

- The fractional-step modularity with second order pressure splitting error;
- Simple mathematical management and matrix-free calculation;
- No limitations on the finite elements.

3.2 Aitken relaxation

The Aitken’s $\Delta^2$ method [64] was extended by other scholars [60, 61] and enjoys immense popularity in accelerating FSI subiterations. We employ this method to deal with the instability caused by the coupling of fluid and structural domains. At each subiteration per time step, the dynamic Aitken factor is estimated by the recursion formula below

$$
\lambda_{iter}^{n+1} = \begin{cases} 
\max(\lambda_{\text{MAX}}, \lambda^n), & iter = 1, \\
-\lambda_{iter-1}^{n+1} \frac{g_{iter}^T (g_{iter} - g_{iter-1})}{\|g_{iter} - g_{iter-1}\|^2}, & iter \geq 2,
\end{cases}
$$

(3.1)

where $\lambda_{\text{MAX}} = 0.1$ and $\lambda_1^0 = 0.5$. The limit of the Aitken factor may be outlined into the range $(0, 1)$ [65].

4 Numerical example

In this section we aim to numerically replicate the experimental investigation conducted by Anagnostopoulos and Bearman [66] where an elastically mounted circular cylinder is allowed to transversely oscillate in the fully laminar flow region. The problem settings are schematically demonstrated in Fig. 4 where $D$ is the diameter of the circular cylinder. The location of the upstream boundary (i.e. the distance between the inlet and the cylinder’s centroid) is $10D$ while that of the downstream boundary is $25.5D$. The spacings between the lateral boundaries and the cylinder’s centroid are both $10D$. The no-slip boundary condition is applied on the cylinder surface. The free-stream velocities are prescribed at the inlet and the pressure-free condition is imposed at the outlet. The velocity-free boundary condition is assigned to both lateral boundaries. All initial variables are assumed to be zero. The physical properties of the problem are set as follows. The mass and diameter of the circular cylinder are $m_2 = 2.979$ g and $D = 0.16$ cm. The spring stiffness $k_2$ and damping factor $c_2$ of the cylinder-spring system are $5790.9$ g/s² and $0.325$ g/s, respectively. The density $\rho$ and viscosity $\mu$ of the fluid are $1.0$ g/cm³ and
0.01 g/(cm · s), respectively. $Re$ ranges from 90 to 130 thanks to various inflow velocities under consideration. The natural frequency $f_{N2}$ of the circular cylinder is 7.016 Hz. The resulting dimensionless parameters are obtained below: the mass ratio $m^*_{2} = 116.37$, the damping ratio $\xi_2 = 1.237 \times 10^{-3}$ and the reduced natural frequency $f_{R2} = 17.961/Re$.

For the sake of computational efficiency, the entire computational domain is divided into three parts: the Eulerian subdomain $A1$, the ALE subdomain $A2$ and the Lagrangian subdomain $A3$ (see Fig. 4 for reference). The size of $A2$ is $6D \times 6D$ while that of $A3$ is $1.2D \times 1.2D$. The points in $A1$ keep fixed at all time while those in $A3$ move along with the circular cylinder. In $A2$ the points are instantaneously updated by MSA. To further lower the numerical cost, some time-invariant matrices in $A1$ are calculated only once at the beginning of the simulation.

The flow past the circular cylinder at $Re = 100$ is evaluated so as to test the mesh sensitivity. We consider three different meshes, i.e., the coarse mesh $M1$ (8092 T3 elements and 4141 points), the medium mesh $M2$ (15856 T3 elements and 8033 points) and the fine mesh $M3$ (31872 T3 elements and 16049 points). The computed response and aerodynamic coefficients of the cylinder are listed in Table 1, including the maximum amplitude $d_{MAX2}$, the mean drag coefficient $C_{D,MEAN}$ and its root mean square (RMS) value $C_{D,RSM}$, the amplitude of lift coefficient $C_{L,MAX}$ and its RMS value $C_{L,RSM}$, Strouhal number $St$ ($St = f_{v}D/U$) and the ratio of vortex-shedding frequency $f_{V}$ to natural frequency $f_{N2}$. It is seen from Table 1 that the deviation is quite small among all mesh resolutions. Thus mesh $M1$ is selected for the sake of accuracy and efficiency. The $M1$ finite element

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$d_{MAX2}$</th>
<th>$C_{D,MEAN}$</th>
<th>$C_{D,RSM}$</th>
<th>$C_{L,MAX}$</th>
<th>$C_{L,RSM}$</th>
<th>$St$</th>
<th>$f_{V}/f_{N2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>0.395</td>
<td>1.869</td>
<td>0.256</td>
<td>0.293</td>
<td>0.179</td>
<td>0.998</td>
<td></td>
</tr>
<tr>
<td>M2</td>
<td>0.398</td>
<td>1.863</td>
<td>0.256</td>
<td>0.297</td>
<td>0.179</td>
<td>0.988</td>
<td></td>
</tr>
<tr>
<td>M3</td>
<td>0.399</td>
<td>1.866</td>
<td>0.257</td>
<td>0.298</td>
<td>0.180</td>
<td>0.999</td>
<td></td>
</tr>
</tbody>
</table>
Figure 5: Mesh and submesh for the transversely oscillating circular cylinder.

Figure 6: Amplitude and frequency ratio of the transversely oscillating circular cylinder.

mesh and the corresponding submesh are plotted in two panels of Fig. 5. The time step is $\Delta t = 1.0 \times 10^{-2}$ and the convergence tolerance is $tol = 1.0 \times 10^{-6}$.

Two vital aerodynamic parameters are the oscillation amplitude $d_{MAX2}$ and the ratio of the vortex-shedding frequency $f_V$ to the natural frequency $f_{N2}$. Both parameters are carefully examined for the circular cylinder transversely vibrating at various $Re$ in Fig. 6.
For the sake of comparison, the well-known Roshko’s Strouhal frequency relationship [67]

\[ St = 0.212 \times \left( 1.0 - \frac{21.2}{Re} \right), \]  

measured for a fixed circular cylinder is superimposed in Fig. 6, where \( St \) is the Strouhal number. Because of the flow conditions and physical properties of the mass-spring system, the circular cylinder will be excited to undergo the frequency lock-in or synchronization.

At \( Re \) below the lock-in region, the oscillations of the cylinder are very faint and the vortices are shedding at the Strouhal frequency with the cylinder vibrating at a frequency lower than its natural frequency. At this point the structural motion is unlocked. The amplitude of oscillation is not modulated until \( Re \) is large enough in this study. Figs. 7(a) and 7(b) show the time histories of the cylinder displacement at \( Re = 95 \) and 99. Apparently, the so-called beating or modulation phenomenon is not observed at smaller \( Re \) but appears at larger one. Near the lower end of the lock-in region, a quite small increase
Table 2: Comparison of results for the transversely oscillating circular cylinder at $Re = 100$.

<table>
<thead>
<tr>
<th>Reference</th>
<th>$d_{\text{MAX}}$</th>
<th>$St$</th>
<th>$f_{V}/f_{N^2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wei et al. [73]</td>
<td>0.442</td>
<td>0.179</td>
<td>0.998</td>
</tr>
<tr>
<td>Schulz and Kallinderis [74]</td>
<td>0.478</td>
<td>0.177</td>
<td>0.987</td>
</tr>
<tr>
<td>Li et al. [75]</td>
<td>0.420</td>
<td>0.179</td>
<td>0.999</td>
</tr>
<tr>
<td>Abdullah et al. [76]</td>
<td>0.290</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Dettmer and Perić [53]</td>
<td>0.397</td>
<td>0.179</td>
<td>0.999</td>
</tr>
<tr>
<td>Yang et al. [77]</td>
<td>0.393</td>
<td>0.178</td>
<td>0.990</td>
</tr>
<tr>
<td>Yang et al. [78]</td>
<td>0.459</td>
<td>0.179</td>
<td>0.997</td>
</tr>
<tr>
<td>He et al. [68]</td>
<td>0.407</td>
<td>0.181</td>
<td>1.01</td>
</tr>
<tr>
<td>De Rosis et al. [51]</td>
<td>0.379</td>
<td>0.167</td>
<td>0.929</td>
</tr>
<tr>
<td>Chern et al. [71]</td>
<td>0.396</td>
<td>0.179</td>
<td>0.997</td>
</tr>
<tr>
<td><strong>Present study</strong></td>
<td><strong>0.395</strong></td>
<td><strong>0.179</strong></td>
<td><strong>0.998</strong></td>
</tr>
</tbody>
</table>

in inflow velocity causes a dramatic raise in amplitude and has the effect of synchronizing the cylinder oscillation frequency and the vortex-shedding frequency. The lock-in phenomenon is evidently observed, starting at $Re = 100$ and ending at $Re = 115$. It is noticed that, the narrower lock-in ranges are computed by our earlier works [19, 20, 68, 69] and other papers [51, 53, 70–72]. Therefore, the present method produces the widened lock-in range and the larger amplitudes at resonance. A few indicators evaluated at $Re = 100$ are listed in Table 2, revealing that the obtained data are in agreement with previously published data [51, 68, 71, 73–78]. Note that lock-in may not be aroused at this given $Re$, such as [66, 72, 79–81]. When lock-in occurs, the vortex-shedding frequency diverges from that predicted by Roshko for a fixed cylinder and becomes identical to the oscillation frequency. In other words, the frequency ratio $f_{V}/f_{N^2}$ roams around unity within the lock-in region, implying the synchronization of the oscillation frequency and the vortex-shedding frequency. This synchronization is responsible for large-scale and drastic motions of the cylinder. We can realize this fact through the time history of the cylinder displacement at $Re = 110$ in Fig. 7(c). The oscillation amplitude abruptly descends to a very low level once $Re$ migrates above the upper end of the lock-in region. The cylinder keeps on imperceptibly oscillating with the growth of $Re$. As $Re$ increases further, $f_{V}$ reaches a large value and coincides with the Roshko relationship well. Again, the modulated amplitude of oscillation is seen at $Re = 120$ in Fig. 7(d) and the cylinder displacement remains feeble therein. Different from [66], the cylinder displacement at $Re$ above the lock-in region is more modulated than that at $Re$ below the region.

The corresponding vorticity fields for three selected Reynolds numbers are illustrated in Fig. 8. The circular cylinder undergoes low-amplitude oscillations outside the lock-in scope, but high-amplitude oscillations within the scope. Unlike [80], all the three vortex-shedding modes behind the cylinder wake are of the $2S$ type. We have never detected the $C(2S)$ mode [82] here. In addition, the vortex spacing is somehow reduced at $Re$ outside
of the lock-in region. The associated modes seem closer to the standard 2S mode for the fixed circular cylinder. This reality is justified by Fig. 6 where the frequency ratio almost tallies with the Roshko curve if Re is not located within the lock-in region.

Our attention is now paid to the oscillation amplitudes of the cylinder computed with and without the SPGP technique. When applying the SPGP technique, the reasonable improvement is achieved with reference to Fig. 9. In particular, we do not receive the larger values of oscillation amplitude at resonance but also obtain a broadened lock-in area. The power spectra of the displacement curve at Re=110 is shown in Fig. 10 by using a fast Fourier transform on the time history of transverse displacement. The normalized value of the oscillation frequency is 0.16296. It is clear that this quantity is as the same as that of vortex-shedding frequency, and is nearly identical to the natural frequency, 17.961/110 = 0.16328. The general features of lock-in are fairly predicted by the present approach.

5 Conclusions

We have attempted in this paper a stabilized second-order AC-CBS-based partitioned semi-implicit FSI coupling strategy by using the ALE finite element formulation. The
quasi-incompressible flows are solved by the second-order AC-CBS scheme in conjunction with the SPGP technique, whereas the structural equation is advanced in time by the Generalized-α method. The ALE dynamic mesh is efficiently updated by means of the combination of MSA and OST-SAM. The partitioned semi-implicit coupling algorithm is formulated in light of the AC-CBS scheme where the AC coefficient is iterated at the implicit stage. It is pointed out that, throughout the entire coupling algorithm, the dual-time stepping is not involved for the solution of quasi-incompressible fluid flows. The main contributions are summarized as follows:

- The fractional-step implementation is extremely simple;
- No algebraic equations are solved and the method is entirely matrix-free;
- Any finite elements are available for the proposed semi-implicit coupling scheme.
The proposed semi-implicit method is validated against the available data for a benchmark FSI problem, i.e. VIV of a transversely oscillating circular cylinder. In contrast with our earlier attempts, the obtained results agree better with those well-documented data. The important lock-in and beating phenomena are detected successfully. The enhanced cylinder amplitude is gained by introducing the SPGP technique into the second-order pressure splitting scheme, although the numerical expenditure is slightly raised. The sensitive pressure estimate in an oscillating cylinder may be stabilized by the present approach.

Acknowledgments

This work is supported by National Natural Science Foundation of China (Grant No. 51508332 and 51508333). The authors thank anonymous referees for their insightful comments in improving this paper.

References


[65] Baek H, Kariadakis GE. A convergence study of a new partitioned fluid–structure interac-


